Modeling of a Horizontal Axis Wind Turbine with Smart Actuators

M.Sc. Thesis

Ben M. Geurts

Delft University of Technology
Faculty of Aerospace Engineering
Department of Wind Energy
2009
## Contents

### Introduction
1. Orientation .................................................. 1
2. Objective ................................................... 3
3. Simulation Options ......................................... 5

### A Aerodynamic Simulation: Vortex Panel Code
4. Basics of Panel Modeling .................................. 10
5. Free-wake Vortex Panel Code in MATLAB ............ 14
6. Geometry Generation ....................................... 16
7. User Input .................................................. 17
8. Data Post Processing ....................................... 18

### B Rigid Geometry Simulation
9. Rotor Geometry ............................................. 21
10. Simulation results .......................................... 23

#### 10.1 Geometrical Flow Angles
11. Force and Moment Distributions ....................... 26
12. Force and Moment Resultants at the Axis of Rotation 33
13. Axial Induced Velocity .................................... 38
14. Wake Visualization ........................................ 39
15. Pressure Distribution ...................................... 40

### C Flaps Affecting Blade Aerodynamics
11. Flap Geometry ............................................. 45
12. Integration in the Simulation ............................ 45
13. Controlled Flap Deflection ............................... 46
14. Effects of Flap Control .................................... 48
15. Effects of Flap Amplitude ................................. 51
16. Applicability of Smart Actuators During Yaw Misalignment 54
17. General Applicability of Smart Actuators ............ 56

### D Influence of Wake Cut-off
18. Limiting Simulation Time ................................. 59
19. Practical Application ...................................... 59
20. Results .................................................... 60
Contents

E  Conclusions 63
   Simulation ................................................................. 65
   Influence of Yaw Misalignment ........................................ 66
   Influence of Flap Deflections ......................................... 66
   Applicability of Flaps on Wind Turbine Blades ................. 66

F  The Next Step: Aero-Elastic Analysis 69
   21  Basics of Aero-Elasticity ......................................... 71
   22  Aero-Elastic Simulation .......................................... 72
List of Symbols

\( \rho \) Air density
\( \Omega \) Rotational velocity of the rotor
\( a \) Induction factor
\( \bar{c} \) Mean chord length
\( c_{M_f} \) Distributed flapping moment coefficient
\( c_{M_{LL}} \) Distributed lead-lag moment coefficient
\( c_{M_p} \) Distributed pitching moment coefficient
\( c_n \) Distributed normal force coefficient
\( c_t \) Distributed tangential force coefficient
\( A \) Rotor swept area
\( C_{M_f} \) Flapping moment coefficient
\( C_{M_p} \) Pitching moment coefficient
\( C_p \) Pressure coefficient
\( C_T \) Thrust coefficient
\( C_Q \) Torque coefficient
\( C_T \) Thrust coefficient
\( F'_n \) Distributed normal force
\( F'_t \) Distributed tangential force
\( M'_f \) Distributed flapping moment
\( M'_{LL} \) Distributed lead-lag moment
\( M'_p \) Distributed pitching moment
\( R \) Rotor radius
\( V \) Local flow velocity
\( V_\infty \) Free-stream wind velocity
1 Orientation

Since ancient times, mankind has been trying to use the immense energy stored in the wind to its advantage. In the evolution of ‘Wind Energy Conversion Systems’, an obvious trend from simple to complex can be seen.

Sails as Energy Conversion
One of the most simple ways to convert wind energy into useful work were the sailboats which cruised the river Nile as early as 5000 B.C. An example of such an ancient sailboat is shown in figure 1. Sails could be operated as simple drag devices with practically no moving parts. In more advanced versions, the sails operate as lift devices, allowing headings almost into the wind direction.

Windmills
A way to convert wind energy into useful work other than transport, at a stationary position, is a lot more complex, even in its most early stages. The first occurrences, very basic systems, were reported around 200 B.C. in China and the Middle East. Simple windmills were extensively used in the food industry until the crusaders brought the idea to Europe around the 11th century. There, the system was refined by the Dutch in order to use wind energy to drain lakes and create new fertile lands (Figure 2(a)).

Wind Turbines
As the industrialization provided powerful steam energy machines, windmills lost their market share for producing mechanical work. But the technological advancements also created a demand for electricity, and windmills turned into wind turbines (Figure 2(b)). The result of this transition is a sudden increase of the complexity
of the whole energy conversion system, now including a lot of moving parts and electronic devices.

**Smart Wind Turbines**

With an increasing complexity, also the risk of failure of a crucial component increases. Current turbines are designed for a 20 year lifetime, and important design requirements are a result of the expected fatigue life of certain subcomponents. Crack growth in materials as a result of fatigue is triggered by a periodic loading of the construction. The number of load-cycles (thus lifetime) and the amplitude of the periodic loading dominate the fatigue behavior. From the S-N-curve in figure 3, it can be easily seen that the expected lifetime greatly increases with decreasing load amplitude.

The conclusion can be quickly made that decreasing or even eliminating the
periodic behavior of the loading can have great advantages, both on the design life, and on the weight of the structure. One of the ways to reduce the periodic behavior of the loading is by applying flaps to the blades of a Horizontal Axis Wind Turbine (HAWT). The flap deflections can be programmed in a smart way to counteract the increasing periodicity of aerodynamic phenomena like wind shear (Figure 4(a)) or yawed operating conditions (Figure 4(b)). A next step adds even more complexity to the system: instead of programming the flap deflections for certain operating conditions, a sensing system can be added, calculating in real-time the required flap deflections. With an autonomous system like this, it would be possible to even use the control surfaces for non-periodic loadings, e.g. occurring due to turbulence and gusts.

2 Objective

This master thesis describes the process of developing a tool that can be used in the design process of a smart rotor system. The tool is built up to analyze a small-scale HAWT rotor with flaps, designed to be tested in the OJF open jet wind tunnel facility at the Delft University Of Technology.

Process Steps

As a first step, an aerodynamic model is used to investigate the loads on the blades of the HAWT in different operating conditions. The background, and the way this aerodynamic model works is explained in part A. The reference loading occurs in pure steady, axial wind, resulting in a steady blade loading. From there on, the loads are analyzed for operations under yaw misalignment conditions, while the wind is kept steady. These situations are described in PART B of this thesis.
INTRODUCTION

Figure 4: Sources of periodic aerodynamic loading

The second step is explained in Part C, and involves adding a time-dependent flap deflection to the steady, axial inflow case. From this step, the effect of the flap deflections on the near wake and the blade loading can be quantified. Combining the knowledge from the yaw misalignment and from the axial case with flap deflections, a first ‘smart’ estimate can be made for the required flap deflections in order to minimize the periodic loadings as a result of the different yaw misalignments.

The complexity of the simulation determines the required time. Therefore it is useful to know what the effect is of using only a limited number of wake-panels to calculate the induced velocity at the rotor-plane. A brief investigation into this effect is done in Part D.

The conclusions that can be drawn from the previous parts are briefly listed in Part E.

Future Steps
In the future, the ‘smart’ part can be integrated within the turbine model, determining the required flap deflection angles on its own at every time step. With the simulation model, the internal controller could be designed before the actual application in the test turbine is done. This can considerably reduce the required wind tunnel test time. No effort has yet been put into this real-time-controller development. With the increasing size of HAWT’s, the elastic effects in large components play a more important role. The aerodynamic loading results in elastic deformations, which in turn change the aerodynamics of the turbine. This vicious circle of changes can have vicious consequences as the most severe effect, flutter, most certainly always results in destruction.
For the small test rotor, the elastic effects are negligible, but when the smart rotor concept is to be applied to large, full scale rotors, the pitching moment produced by the flap deflection can trigger unwanted aero-elastic phenomena. To avoid unwanted behavior, the aerodynamic model can be expanded with an elastic analysis part, calculating the deflections of the blades under the aerodynamic loading. The basis for this procedure is already made, and is explained in Part F.

3  Simulation Options

For the aerodynamic analysis of the wind turbine rotor, there are several modeling options available. Every model has advantages and disadvantages. The main options, in order of complexity, are the Actuator Disc Theory, Blade Element Momentum Theory (BEM), Vortex Modeling and Computational Fluid Dynamics (CFD).

3.1  Actuator Disc Theory

The Actuator Disc Theory is the simplest approximation of flow effects in the rotor disc. As explained in [7], [3] and [6], the rotor is assumed to be a uniform circular disc through which air can flow. At the location of the disc, energy is extracted from the flow in the form of forces on the disc, and counteracting forces on the flow. A local pressure difference is given between the two sides of the disc, as is shown in figure 5. The flow is slowed down, and from the conservations of mass, momentum and energy, the forces can be calculated giving an indication of the rotor forces. However, as also mentioned in [7], the actuator disc theory can only yield integrated values of the actuator disc flow parameters, like the average flow velocity through the disc as a function of thrust coefficient.

![Figure 5: Actuator Disc Theory: Axial velocity and pressure across the rotordisc.](image)

For the purpose of investigating the effects of flap deflections on the near wake structure, and on the blade loading, this method is unsuited, as it can provide neither load distributions over the length of the blade, nor near-wake behavior.
3.2 Blade Element Momentum Theory

BEM theory is probably the most widely applied technique, as it provided very fast results, with reasonable accuracy. The rotor disc, and at the same time the whole stream-tube containing the rotor disc, is split-up in discrete radial stream-tube elements. From the forces on the blade element at a selected radial position, the induced flow velocity is derived. Combining this induced velocity with the blade and free-stream velocities, the local angle of attack of the blade element can be found. With the help of empirical 2D airfoil data, the local velocity and angle of attack can yield new values for the forces on the blade element. This iterative process can be repeated until a steady solution is obtained. As stated by Hansen et al. in [3], one of the main disadvantages of this method is that the accuracy of the results depends on the accuracy of the 2D airfoil data, applied to an intrinsic 3D flow field. Also, the BEM theory can provide fast and accurate results but these are steady, or at most quasi-static, which reduces its applicability when looking at time-dependent flap control. Nevertheless, BEM can be a time-efficient option for returning reliable force distributions over the whole blade, but it falls short when the near-wake flow behavior is under investigation.

3.3 Vortex Modeling

Vortex models use the concept of vorticity transport to calculate the influence of the wake of a blade on the inflow conditions. With those inflow conditions known, the sectional lift and drag of the blade can be derived. Using the Kutta-Joukowwsky theorem, the shed vorticity can be derived from the lift of the blade section at every time-step. The shed vorticity, in the form of vortex rings, together forms the wake of the rotor-blade. The blade can be represented as a lifting line containing a vortex counteracting the sum of all the shed vorticity (As explained by K.R.Dixon in [4]). The disadvantage of this model can be found in the calculation of the lift and drag over the blade. As in the BEM theory, the forces on the blade are obtained using look-up tables containing 2D airfoil data. Thus the accuracy of the model is dependent on the accuracy of the 2D data, combined with a inherently 3D flow over the blades. On the other hand, the model can provide information about the shape of the near-wake, which is a big advantage.

3.4 Panel Method

The way in which a panel method handles the wake of a rotor-blade is the same as in the vortex method, i.e. the shed vorticity is stored in the wake, and provides insight in the near-wake structure. At the same time, the disadvantages of the vortex model are avoided by eliminating the lifting line concept. In the panel method as explained and used by Kristian Dixon [4], the blade surface is represented by a number of
discrete panels, as shown in figure 6. This panel-representation of the blade also has the great advantage that it uses an accurate representation of the exact geometry as would occur in the real life rotor. Linked to this, it incorporates three dimensional flow phenomena. However, the great advantage comes accompanied with a great drawback: Viscous effects are omitted. As a result, separated flow and dynamic stall cannot be modeled with the panel code.

![Figure 6: Panel method: Discretised representation of an airfoil section and the shed wake.](image)

### 3.5 Computational Fluid Dynamics (CFD)

With increasing computational power, the application of CFD solvers gets more widespread. The behavior of all fluids can be harnessed in the Navier-Stokes equations. These equations describe the balance between the momentum, the viscosity and the pressure within the fluid. CFD packages solve these very complex Navier-Stokes equations in a simplified form in order to find a solution for the fluid surrounding the body. Mostly, the equations are used in the form of the Reynolds Averaged Navier-Stokes (RANS) form. The Reynolds averaging gives a time-averaged solution which can be obtained when the turbulence behavior of the flow described in advance. Turbulence models are typically tuned for particular applications such as problems with boundary layers, shear layers, or mixing layers around jets. Of course, applying a different turbulence model will give a different solution of the flow situation in the end. According to Vermeer et al. in [7], the advantage of CFD is obviously the power to capture and include nearly all flow phenomena. But the advantage comes at a high price, as CFD calculations of complex flow situations require vast amounts of computation effort. On top of that, the reliability of the obtained results strongly depends on the reliability and applicability of the turbulence model used.
Part A

Aerodynamic Simulation: Vortex Panel Code
4 Basics of Panel Modeling

In his Master thesis, Kristian Dixon [4] neatly explains the theoretical background on which a panel code is based. In this section, this background theory is briefly summarized. For a more elaborate explanation, the reader should refer to [4].

4.1 The Navier-Stokes and Laplace Equations

The behavior of all fluids can be expressed as a series of coupled, non-linear partial differential equations known as the *Navier-Stokes* (N-S) equations. This system of equations represents the conservation of mass, momentum and energy (equations A-1, A-2 and A-3 respectively). They hold all fundamental physics on which aerodynamics is based. The N-S equations are most easily understood when expressed in their integral form which follows directly from conservation laws. Via the divergence theorem, they can also be transformed into a differential form:

\[
\frac{D\rho}{Dt} + \rho \nabla \cdot \vec{q} = 0 \quad (A-1)
\]

\[
\rho \frac{D\tilde{q}_i}{Dt} = \rho f_i + \frac{\partial \tau_{ij}}{\partial x_j} \quad (A-2)
\]

\[
\frac{D(e + \frac{1}{2}V^2)}{Dt} = \tilde{q}_{flux} + \frac{1}{\rho} \left[ \nabla (k \nabla T) - \nabla (p \vec{q}) + \rho \frac{\partial (\tilde{q}_j \tau_{ij})}{\partial x_i} \right] \quad (A-3)
\]

In order to obtain computationally more feasible equations, several simplifications are required. As the operating conditions of a HAWT feature relatively low speeds (Mach Number $M < 0.3$), and typical Reynolds numbers in the range $10^5$ to $10^6$, the flow can be assumed to be incompressible, adiabatic and inviscid, everywhere outside of the boundary layer. By neglecting the effects of compressibility, heat transfer and friction, equations A-4 and A-5 can be formed.

\[
\frac{\partial \tilde{q}}{\partial t} + \tilde{q} \cdot \nabla \tilde{q} = \tilde{f} - \frac{\nabla p}{\rho} \quad (A-4)
\]

\[
\nabla \cdot \tilde{q} = 0 \quad (A-5)
\]

Equation A-4 is known as the Euler Equation, and describes the relation between the local flow velocity and the local fluid pressure. The velocity field in equation A-5 can also be expressed as a velocity potential (equation A-6). This way, equation A-5 transforms into the Laplace equation (A-7).

\[
q = \nabla \Phi \quad (A-6)
\]

\[
\nabla^2 \Phi = 0 \quad (A-7)
\]
This Laplace equation is a linear differential equation, which implies that any two solution for the problem can be added up and will together provide a new solution. This property is extremely useful when the solution of a given flow situation is to be expressed as a combination of fundamental solutions.

4.2 Singular Solutions of the Laplace Equation

The solution for the flow around a given body can be described as the addition of a number of separate solutions thanks to the linearity of the Laplace equation. There are several useful flow modeling solutions possible. The solutions described here are solutions with a singularity at their origin. This means that they can provide a flow solution at all points in the (quasi-) infinite domain except at their center-point.

- **Source**
  
  A singular point from which flow emanates in all directions is called a source (figure 1(a)). The velocity of the flow resulting from a source point is related to the distance from the source point with $\frac{1}{r}$ in 2D and $\frac{1}{r^2}$ in 3D. When the strength of a source is negative, it is a point where the flow is drawn to from all directions, and is called a **sink**.

- **Doublet**
  
  When a source and sink of the same strength are positioned infinitely close to each other, a doublet is formed (figure 1(b)). The amount of mass generated by the source is absorbed by the sink, and an asymmetric flow pattern results. Unlike the source and sink, a doublet not only has a position and strength, it also has an orientation.

- **Vortex Point**
  
  Unlike the sources, sinks and doublets, no flow is produced or eliminated as a result of the presence of a vortex. Instead, the vortex produces a rotational velocity profile around its center (figure 1(c)). As in the case of the source and sink, the velocity relates to the radial distance to the center.

4.3 3D Constructions

The source, sink, doublet and vortex singularities mentioned above are 2D flow objects. For the simulation of a 3D flow, different 3D constructions can be produced with these singularities. Surface panels can be built up as a continuous distribution of singular points. The line element in figure 1(d) is the 2D-equivalent of a 3D doublet panel. With these different flow elements, it is possible to create a geometry around which air flows without entering the body, creating a representation of the actual (inviscid) flow around a real body immersed in a flow. There are several ways to use the singularities for arriving at a problem that can be solved numerically. They are summarized as follows:
PART A. AERODYNAMIC SIMULATION: VORTEX PANEL CODE

Figure A-1: Singular flow elements

- **Source Doublet formulation** (lifting surfaces with thickness)
- **Source Neumann formulation** (non-lifting surfaces with thickness)
- **Doublet Neumann formulation** (lifting surfaces with zero thickness)

There are also two ways of enforcing the boundary condition on the surface:

- **Dirchlet problem** the internal potential is specified at (or very near) the boundary – this boundary condition is only applied to those problems where the geometry has appreciable thickness.

- **Neumann problem** the normal velocity is specified on the boundary.

The majority of panel methods for lifting flows with thickness utilize the source doublet formulation with the Dirchlet boundary condition because it affords a simpler implementation.

In applications where the geometry has appreciable thickness and generates lift there are many source and doublet-distributions which will satisfy the boundary conditions. An ancillary condition must be imposed in order to find a unique solution that is representative of physical flows. This is known as the ‘Kutta Condition’.

### 4.4 The Trailing Edge Kutta Condition

The Kutta Condition requires that the flow leaves the trailing edge in a smooth continuous fashion and that the velocity there is finite. If the trailing edge has a finite angle, the normal and tangential component of the velocity must be zero, resulting in a stagnation point. If the trailing edge is represented by a cusp, i.e. the angle between upper and lower surface is zero, the edge point does not necessarily need to be a stagnation point and a finite tangential velocity component can exist.
The normal component, however, should be zero. In general, there should exist no pressure difference between the upper and lower surface at the trailing edge and no pressure difference over the wake surface. A sharp trailing edge is a region of infinite curvature, and for the flow to move around such a geometry would require infinite velocities which are clearly non-physical. Doublet and Vortex singularities may be thought of as being responsible for turning the flow. This means that the Kutta Condition implies that the circulation at the trailing edge must be zero (A-8).

\[ \gamma_{\text{TrailingEdge}} = 0 \] (A-8)

The enforcement of this Kutta condition depends on the numerical scheme. In some cases, like a 2D lifting surface represented by discrete vortices, the Kutta condition is automatically satisfied. This set-up will automatically have zero circulation at the trailing edge. When this scheme is interpreted in 3D, its equivalent is a horseshoe-vortex representation of a wing. The horseshoe is a actually a vortex ring with three elements influencing the airflow over the wing, and the fourth element left at an infinite distance behind the wing.

4.5 Simulation Elements

The surface of the simulated immersed body is composed of discrete panels consisting of source-doublet distributions. The Kutta condition prescribes the strength of the shed wake panels at the trailing edge. In the wake, the requirement that there is no normal velocity across the panels is not present. As a result, the source distribution is nog needed. When looking at the velocity induced by the doublet distribution in figure 1(d), it can be understood that this velocity profile is equivalent to the profile induced by two singular point vortices with opposite strength at the end points of the line element. This analogy also holds in 3D, where a doublet panel can be represented by a vortex ring given by the edges of the panel. The advantage with this alternative representation is the fact that the vortex ring can deform in any shape while still maintaining the ring properties.

The extended derivation of the formulae for the panel code used in this thesis can be found in [4].
5 Free-wake Vortex Panel Code in MATLAB

The selection of a simulation method was simplified by the availability of a vortex panel code created by Kristian Dixon in the framework of his Msc research [4]. His code was written in MATLAB as a means to investigate the near-wake structure of a Vertical Axis Wind Turbine (VAWT). The code has dual functionality, being able to handle both 2D and 3D cases. For the simulations in this thesis, Kristians existing Panel code is altered and optimized to investigate the near-wake and load cases of a HAWT. For this purpose, the 2D-capability is rendered useless, so all changes are specifically designed for the 3D-HAWT situation.

The code makes use of MATLAB’s Object Oriented Programming. This means that during the simulation, the data is stored in a tree-structure with different object levels. The lowest level is named ‘B’, and is a Body object. It contains all information about a single geometrical structure, like a single rotor-blade of a HAWT. There can be several bodies having the same movement pattern. These bodies are grouped in an Ensemble object (‘E’). They are linked together at a fixed orientation and position, and the motion function for the ensemble is applied to all bodies contained in this ensemble. Thus, the complete rotor is stored as an ensemble, and contains the two rotor-blade bodies. The trunk of the tree is a variable named ‘S’, and it is a Simulation object. Within the simulation object, it is possible to have several ensembles. But for this simulation, there is only one ensemble, the rotor. It could be possible, however, to add an extra ensemble for the support tower and the rotor-hub.

The general layout of the calculation procedure is shown in the flow-diagram in figure A-2. The simulation process consists of several distinct parts. The first part shown in figure A-2 is the User Input part. This is a pre-simulation stage, manually performed by the user. The user should select settings, and edit geometry variables to specify the situation to be simulated.

Once the MATLAB-run is started, the computer performs the next stages of the simulation by itself. The first automated stage is the phase in which MATLAB builds the simulation file. It starts by gathering all needed information from the simulation input provided by the user. This input specifies which ensembles E are needed to build the simulation S. The ensembles consist of several bodies B which are pointed out in the ensemble input file. Finally, the body input describes the exact geometry of the rotor blades. The actual generation of the blade geometry is done in the geometry building file, which employs the available blade data files. The contents of the various input files are explained in section 7.

At this point the simulation is set to take off. The run starts to analyze the aerodynamics of the rotating rotor. While all the preceding activities occur only once per simulation, the next step is repeated for every specified time-step. The internal computation procedure of the solving step of the simulation is explained in the flow
diagram in figure A-3. The most important step within this flow diagram is the *Update Geometry* step. During this step, the geometry is deformed. This process will first deform the panels making up the flaps according to the required flap angle at the current time-step. Then the whole blade is rotated into the required current azimuthal position.

Figure A-2: Overall Flow Diagram of the Panel Code

Figure A-3: Flow Diagram of the Time-loop within the Simulation
6 Geometry Generation

The Vortex code is built up in such a way that the geometry can be summarized as a collection of points in space (VertPts) and a set of panel indices (PanelInds). The VertPts is a collection of the X, Y and Z-coordinates of every point in the inertial reference frame. The PanelInds contains four values for every panel. The four values indicate which four points of the VertPts matrix make up a panel.

When constructed in the right way, VertPts and PanelInds can provide ALL necessary geometrical data for the objects in the simulation.

The means to build these two important matrices was originally left open for the user to define. In the original code a general constructor was present. This constructor generated a straight wing with constant airfoil cross-section, which was simply specified by a NACA 4-digit airfoil code. For its initial purpose, the investigation into flow phenomena of Vertical Axis Wind Turbines, this setup was sufficient. But when adapting the code to simulate a real-case HAWT, twist and taper are essential geometrical features. In most cases also the airfoil cross-section changes along the blade span, and will hardly ever consist of standard NACA airfoils. Therefore the new builder for HAWT-geometry has to be flexible considering selected twist, taper and airfoil-sections, while always setting up the VertPts and PanelInds matrices in the proper way. On top of that, it must be possible to specify the different inputs in an easy and transparent way.
7 User Input

Within the program’s main folder, there are two folders in which input-data for the simulations is stored. The first one, *BladeData*, contains txt-files with all the needed blade geometry data for all simulations. The second one contains the MATLAB-files with simulation-specific inputs, and is named according to the specific simulation.

**BladeData Folder**

还算**ChordDistribution.txt** contains the chord length in meters for at least two (tip and root) blade sections. The eventual chord distribution is obtained using a MATLAB spline interpolation. The blade span positions are to be given as non-dimensional values from 0 (at the blade root) up to 1 (at the blade tip) as is also used in table B-2.

还算**TwistDistribution.txt** is similar to the ChordDistribution-file, but now contains the local twist angle of the blade in positive degrees nose-up.

 Allan**PitchAngle.txt** only holds one single number: the fixed pitch angle in positive degrees nose-up.

还算**AirfoilData.txt** comprises the 2D coordinates of the airfoils used. There can be different files, one for every needed airfoil. The coordinates are assumed to be made non-dimensional (chordwise coordinates range from 0 to 1)

**Simulation Folder**

还算**INPUT_SimulationSetup.m** contains variables that define the specific simulation (E.g. the number of time-steps) and variables that define how the program should behave in certain circumstances (E.g. selection of vortex core approximation method). The latter are rarely changed, in order to be able to compare the results of different simulations.

总算**INPUT_EnsembleSetup.m** stores the information about which bodies are connected in the ensemble, and what their relative position is. It also contains the link to the motion-function which determines a pre-described motion for this ensemble.

还算**INPUT_BodySetup.m** holds part of the body-specific data like the number of discretised panels. Additionally, it references to the *WingBuilder.m* for the actual generation of the body geometry.

**GeometryBuilder.m** is the heart of the geometry generation, as it builds the *VertPts* and the *PanelInds* from the given airfoil, twist and chord distributions in the BladeData folder.
8 Data Post Processing

When the simulation is finished, the next step is to extract the data from the saved simulation file. This process happens in 2 steps. First, a stand-alone file is run which loads the simulation, and calculates the wanted variables. All the processed data is saved in a single post processed data-file: PPData.mat the purpose of this procedure is that the postprocessing can be performed on a computer without display (E.g. a dedicated cluster) and then later, the plots can be produced from only the PPData-file, without requiring to load the large simulation files. The aerodynamic properties of the rotor which are of interest for this thesis are mentioned in table A-1.

Table A-1: Investigated Aerodynamic Properties

| Angle of Attack & Local Inflow Angle |
| Induced Velocity                  |
| Span-wise Force and Moment Distributions |
| Root Moments & Power              |
| Chord-wise Pressure Distributions |
| Tip Vortex Locations              |

Of course not all of the data extracted from the simulation is of equal importance in the framework of this thesis. A number of the calculated aerodynamic effects are only produced in order to have a general reference frame when the simulated rotor is tested in the wind tunnel. Among these are the tip vortex locations, the rotor torque and power output. In Part C, the effect of flaps on the aerodynamics of the rotor is investigated. For this part, the load distributions on the blades can be of particular interest.

Part D looks into the effect of including only a limited number of wake panels in the calculations. For this purpose, the change of induced velocity as a result of the varied wake length is looked at.
Part B

Rigid Geometry Simulation
9 Rotor Geometry

From research point of view, the best way to validate a simulation effort is to compare it with real test results. Tests with controlled and isolated flow phenomena are hardly ever conducted on full scale rotors. Thus, for comparison purposes, it is chosen to directly simulate the scaled test rotor instead of its full scale equivalent. The test rotor is a simple 2-bladed rotor developed at the Delft University of Technology for testing in its Open Jet Wind Tunnel Facility (OJF). The data describing the rotor geometry is given in table B-1. The twist distribution and chord length distribution are shown in table B-2. A 3D plot of the rotor-blades is shown in figure B-1. In this plot, the rotor-blades are already discretised using flat panels as is used in the panel method. The rotor is positioned with respect to the inertial reference frame as it would be positioned during the tests in the wind tunnel. The origin of the system is located at the base of the rotor tower with the wind blowing in Y-direction.

Table B-1: Rotor Characteristics for the OJF-rotor

<table>
<thead>
<tr>
<th>Rotor Characteristic</th>
<th>value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of blades</td>
<td>2</td>
</tr>
<tr>
<td>Radius</td>
<td>0.9m</td>
</tr>
<tr>
<td>Tip speed ratio</td>
<td>4</td>
</tr>
<tr>
<td>Pitch Angle</td>
<td>0°</td>
</tr>
<tr>
<td>Airfoil</td>
<td>DU96–W–180</td>
</tr>
</tbody>
</table>

Table B-2: Chord and Twist distributions of the OJF-Rotor.
The span position is zero at the blade root, and one at the blade tip

<table>
<thead>
<tr>
<th>Span Position [-]</th>
<th>Chord-Length [m]</th>
<th>Twist Angle [°]</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0.3000</td>
<td>-17</td>
</tr>
<tr>
<td>0.0795</td>
<td>0.2673</td>
<td>-12.9258</td>
</tr>
<tr>
<td>0.1591</td>
<td>0.2373</td>
<td>-9.8584</td>
</tr>
<tr>
<td>0.2386</td>
<td>0.2100</td>
<td>-7.3974</td>
</tr>
<tr>
<td>0.3584</td>
<td>0.1739</td>
<td>-4.4187</td>
</tr>
<tr>
<td>0.4781</td>
<td>0.1439</td>
<td>-2.0151</td>
</tr>
<tr>
<td>0.5979</td>
<td>0.1200</td>
<td>0</td>
</tr>
<tr>
<td>0.7319</td>
<td>0.1200</td>
<td>0</td>
</tr>
<tr>
<td>0.8660</td>
<td>0.1200</td>
<td>0</td>
</tr>
<tr>
<td>1</td>
<td>0.1200</td>
<td>0</td>
</tr>
</tbody>
</table>
Figure B-1: Rotor Setup
10 Simulation results

This section gives a summary of all plotted data for the standard situation as a reference condition for the evaluations in the following parts. This rigid blade simulation features both pure axial inflow and yawed inflow conditions, without any motion of the control surfaces.

10.1 Geometrical Flow Angles

- The azimuth-dependent Inflow Angle (figure B-2) is calculated combining the wake-induced velocity in the 3/4-chord point and the rotational component of the blade velocity for a number of radial positions. It should come as no surprise that the inflow angle near the root (18% of the radius) is a lot larger than that near the tip (99% of the radius). In figure 2(b) the periodic effects due to the yawed inflow condition can clearly be seen as a cyclic variation of inflow angle. This results from the summation of the free-stream wind velocity with the rational velocity of a local blade section.

- The Angle of Attack (figure B-3) is derived by adding up the local twist angle (nose-up is positive) with the local Inflow Angle from figure B-2. The data in table B-2 shows that the twist angle at the outboard section of the rotor-blades is zero. Therefore, the local angle of attack at 99 and 77% of the radius is exactly the same as the local inflow angles shown in figure B-2. The angle of attack at the root section in the axial inflow case is well over 35 degrees, and in the yawed inflow conditions, the root angle of attack peaks well over 45 degrees. These values predict severely stalled flow conditions. But since one of the main disadvantages of this panel code is the inability to handle stalled flow, the effects of the stalled flow, and the inaccuracies introduced by its absence, remain hidden until this exact rotor shall be tested in the wind tunnel. Looking at the tip section, an even more disturbing conclusion can be drawn. The local chord length at the tip, combined with the local inflow velocity gives a tip Reynolds number of around 400000. When analyzing the used airfoil as a 2D airfoil section using XFOIL, the Cl-\(\alpha\) curve and pressure distribution in figures 5(b) and 5(a) are obtained. From these figure, it becomes clear that at an angle of attack of 14°, the flow over the upper side of the airfoil is separated from mid-chord. The transition point on the upper surface where the flow transforms from laminar to turbulent according to XFOIL lies at about three percent of the chord. Normally, when a forced transition point is moved forward, the occurrence of stall is delayed. But in this case the transition point is already almost at the leading edge of the airfoil, and still severe stall occurs over the aft part of the blade. This has as a consequence that the flaps will operate in constantly separated flow. Therefore, the quantitative results in this investigation need to be treated with extreme caution.
• In yawed flow conditions, the local angle of attack varies with azimuthal position. The **Variation of Angle of Attack** along the blade span is shown in figure B-4.

![Variation of Angle of Attack](image)

(a) Axial Inflow  
(b) Yawed Inflow (15°)

**Figure B-2: Inflow Angle vs Azimuthal Position**

![Angle of Attack](image)

(a) Axial Inflow  
(b) Yawed Inflow (15°)

**Figure B-3: Angle of Attack vs Azimuthal Position**
10. Simulation results

Figure B-4: Angle of Attack Variation vs Blade Span-wise Position

(a) Axial Inflow

(b) Yawed Inflow (15°)

Figure B-5: 2D analysis of the DU96-W-180 airfoil using XFOIL ($Re = 400000$)

(a) Chordwise distribution of pressure coefficient. $\alpha = 14^\circ$ (at the tip)

(b) Lift coefficient vs angle of attack
10.2 Force and Moment Distributions

- The Normal Force Coefficient \( (c_n) \) along the blade span is plotted in figure B-6 as a function of azimuth angle. It is the axial force component, positive in the direction of the wind. The value plotted here is based on the distributed normal force \( F'_n \) in Newton per meter of span, made dimensionless using the air density \( \rho \), the free-stream wind velocity \( V_{inf} \) and the mean chord length of the blade \( \bar{c} \) according to equation B-1.

\[
c_n = \frac{F'_n}{\frac{1}{2} \rho V_{inf}^2 \bar{c}}
\]  

\[(B-1)\]

(a) Axial Inflow  
(b) Yawed Inflow (15°)

Figure B-6: Normal force coefficient for all azimuth and span-wise positions

Figure B-7: Span-wise distribution of the mean normal force coefficient (Axial inflow)
For clarity, the mean value of the span-wise distribution is given in figure B-7. This plot can be seen as a cross-section of the polar plot from the center outward.

The span-wise profile of the normal force coefficient in figure B-7 has a shape which can be expected. The only unexpected behavior that shows up is the drop to negative values near the root of the blade. This behavior can not directly be explained by evaluating the physical flow behavior. In all cases, the normal force is expected to be purely positive, i.e. in the direction of the wind.

The explanation of this phenomenon lies in the **3D-effects** of several geometrical properties. The unexpected force values originate from the high twist angle at the root sections, combined with a rather large chord length at a low radial distance. But including 3D-effects is one of the main strengths of the panel code, then why does this strange phenomenon occur? The negative forces do not occur due to a calculational mishap, or a wrong simulation setup. It is a ‘simple’ but definitely unexpected result of the way the forces on the local panels are added up to give the resultant force for that specific airfoil section: the panel contributions are added up per 2D section of the blade. As can be seen in the pressure distribution shown in figure 5(a), the force is obtained as a delicate balance between the local pressures on the top side (under-pressure) and on the bottom side (over-pressure).

Actually, the forces for an airfoil section are obtained as the summation of the ‘under-pressure’ and the ‘over-pressure’ multiplied with their respective local panel areas. Further away from the center of rotation, and thus also closer to the trailing edge of the root section, the local velocity is higher, and thus the local pressures are smaller (i.e. under-pressure becomes stronger and the over-pressure becomes weaker). Since the panel surface area which is located close to the trailing edge is larger on the bottom surface of the airfoil than it is on the upper surface (figure 8(b)), the effect of the locally reduced pressure acts more on the bottom surface than it does on the top surface of the airfoil section. As a result, the delicate pressure-times-area–balance is disturbed.

So in the end, the force contributions on the panels are not wrong, nor are they misinterpreted. They simply add up in a different way than would initially be expected. An alternative would be to do the summation of the force contributions along a line of constant radius, rather than along an airfoil section. In that case, the span-wise plot of distribution of the normal force coefficient would show different behavior, without negative force components. The resulting forces at the sections close to the root radius would be reduced, and those sections close to the exact radial position of the root trailing edge would...
be increased due to the longer virtual chord length.

**Although the obtained results seem unrealistic, they are in fact perfectly normal.**

Once this is known, there is no need to try to change the way the panel contributions to the force are added up. In analogy with the reasoning above, also the negative values in the force and moment plots on the following pages can be explained.

\[
c_t = \frac{F'_t}{\frac{1}{2} \rho V^2 \infty} \tag{B-2}
\]

Figure B-8: Geometrical 3D-effects
- The **Tangential Force Coefficient** ($c_t$) is shown in figure B-9. The distributed tangential force per meter of span ($F'_t$) is calculated as the component in the rotor plane, and is positive in the direction of the rotational velocity of the rotor. It is made dimensionless using equation B-2. The mean value of the tangential force distribution is given as a function of span-wise position in figure B-10. Apart from the negative values near the root (which are explained on page 27), the plotted results are neatly confirming expected values. As expected under yawed inflow conditions, there is a clear shift in azimuth of about $-45^\circ$ of the maximum and minimum loads with respect to the zero and $180^\circ$ positions respectively. This phase-shift will also appear very clear later-on in the analysis of the rotor torque in figure B-20 on page 35.

![Figure B-9: Tangential force coefficient for all azimuth and span-wise positions](image)

![Figure B-10: Span-wise distribution of the mean tangential force coefficient (Axial Inflow)](image)
The Pitching Moment Coefficient ($c_{M_p}$) shown in figure B-11 is the non-dimensionnal equivalent of the local pitching moment along the blade span ($M'_p$), using equation B-7. The mean span-wise distribution of the pitching moment coefficient is shown in figure B-12. In these plots, there is a clear kink around 70% of the radius. There is no real discontinuity in the geometry. Instead, this kink is caused by the transition from a blade with variable twist and variable chord on the inboard part, and a straight untwisted wing on the outboard sections.

$$c_{M_p} = \frac{M'_p}{\frac{1}{2} \rho V^2 \ell^2}$$  \hspace{1cm} (B-3)

Figure B-11: Pitching moment coefficient for all azimuth and span-wise positions

(a) Axial Inflow  \hspace{1cm} (b) Yawed Inflow (15°)

Figure B-12: Span-wise distribution of the mean pitching moment coefficient
10. Simulation results

- The non-dimensional **Flapping Moment Coefficient** \( c_{M_f} \) is given in figures B-13 and B-14. The plotted values can be interpreted as the contributions of local airfoil sections to the total root flapping moment. This is a straightforward multiplication of the local axial force with the local radius. The non-dimensionality is obtained by applying equation B-6. As can be seen in figures B-6 and B-13, the effect of the yaw misalignment on the axial force and the flapping moment is a lot less than the effect on the tangential force components, and thus also the lead-lag moment and the rotor torque.

\[
c_{M_f} = \frac{M'_f}{\frac{1}{2} \rho V^2 \pi \epsilon^2}
\]  

\[(B-4)\]

![Figure B-13](image1.png)

(a) Axial Inflow  
(b) Yawed Inflow (15°)

Figure B-13: Flapping moment coefficient for all azimuth and span-wise positions

![Figure B-14](image2.png)

Figure B-14: Span-wise distribution of the mean Flapping moment coefficient
PART B. RIGID GEOMETRY SIMULATION

- The non-dimensional **Lead-Lag Moment Distribution** \( c_{M_{LL}} \) is given as a function of azimuth angle in figure B-15. The plotted values can be interpreted as the contributions of local airfoil sections to the total rotor torque. The non-dimensional moment coefficient is obtained by applying equation B-5.

\[
c_{M_{LL}} = \frac{M'_{LL}}{\frac{1}{2} \rho V^2_{\infty} \tau^2}
\]

(B-5)

Figure B-15: Lead-Lag moment coefficient for all azimuth and span-wise positions

Figure B-16: Span-wise distribution of the mean Lead-Lag moment coefficient
10.3 Force and Moment Resultants at the Axis of Rotation

The resulting forces and moments at the center of rotation can simply be obtained by integrating the local force components from section 10.2 over the blade span. The resulting forces and moments are made non-dimensional using the air density $\rho$, the free-stream wind velocity $V_\infty$, the swept area of the rotor $A$, and the rotor radius $R$ according to equations B-6 until B-9. The power coefficient is obtained from the rotor torque according to equation B-10 where $\Omega$ is the rotational velocity of the rotor.

Both the total flapping and the total pitching moment are interesting for the structural design of the blades, and are therefore given as totals per blade. The thrust, rotor torque and power of the turbine are most useful when they are given as totals for the whole rotor, as a summation of the different blade contributions. However, the target of this thesis is to focus on the occurrence of periodic effects. These periodic effects of the yaw misalignment and the flap deflections are almost not present anymore in the total rotor characteristics like power, as a result of the rotor having two blades. In figures B-21 and B-22, this can be seen clearly for the rotor power. For the flap deflection case this phenomenon is similar, as the flap deflection patterns of both blades are the identical. For this two-bladed rotor, when one flap is deflected upward, at the same time, the other flap is deflected equally much downward. When thrust, torque and power contributions of the blades are added up, the effect of one flap cancels out the effect of the other.
PART B. RIGID GEOMETRY SIMULATION

- **Flapping Moment**

  ![Flapping Moment Diagram](image)

  Figure B-17: Root flap-wise bending moment (non-dimensional) vs. azimuth

- **Pitching Moment**

  ![Pitching Moment Diagram](image)

  Figure B-18: Root pitching moment (non-dimensional) vs. azimuth
10. Simulation results

- **Rotor Thrust**

  ![Rotor Thrust vs. azimuth (contribution of one single blade)](a) Axial Inflow  
  
  ![Rotor Thrust vs. azimuth (contribution of one single blade)](b) Yawed Inflow (15°)

  Figure B-19: Rotor Thrust vs. azimuth (contribution of one single blade)

- **Rotor Torque**

  ![Rotor torque vs. azimuth (contribution of one single blade)](a) Axial Inflow  
  
  ![Rotor torque vs. azimuth (contribution of one single blade)](b) Yawed Inflow (15°)

  Figure B-20: Rotor torque vs. azimuth (contribution of one single blade)
PART B. RIGID GEOMETRY SIMULATION

- Power

![Graphs showing rotor power vs. azimuth for axial inflow and yawed inflow (15°).](image1)

Figure B-21: Rotor Power vs. azimuth (contribution of one single blade)

![Graphs showing rotor power vs. azimuth for the total rotor.](image2)

Figure B-22: Rotor Power vs. azimuth (the total rotor)

Since the plots in this section are merely integrations of the distributed versions in section 10.2, the phenomena which occur can be seen as a summary of the phenomena shown in the distributed cases. The $-45^\circ$ phase-shift of the more or less sinusoidal behavior of the loads with respect to the sinusoidal behavior of the yaw misalignment reappears in pretty much all the above plots.
The most important observation that can be made from the above plots is the sensitivity of the rotor torque and thus power with respect to the yaw misalignment compared to that of the other blade loads. Of course, when the summation is made to obtain the total rotor power, the periodic effect disappears. But in the case of e.g. a three-bladed rotor, the periodic effect will still be strongly present in the power output. From this point of view, it might be preferable to tune the control of the flap in order to minimize the amplitude of the power variation rather than optimizing for the elimination of the periodic behavior in e.g. the root flapping moment.

\[ C_{M_f} = \frac{M_f}{\frac{1}{2} \rho V_\infty^2 AR} \]  \hspace{1cm} (B-6)

\[ C_{M_p} = \frac{M_p}{\frac{1}{2} \rho V_\infty^2 AR} \]  \hspace{1cm} (B-7)

\[ C_T = \frac{F_T}{\frac{1}{2} \rho V_\infty^2 A} \]  \hspace{1cm} (B-8)

\[ C_Q = \frac{F_Q}{\frac{1}{2} \rho V_\infty^2 AR} \]  \hspace{1cm} (B-9)

\[ C_P = C_Q * \Omega \]  \hspace{1cm} (B-10)

\[ a = \frac{V_\infty - V_{local}}{V_\infty} = \frac{V_{induced}}{V_\infty} \]  \hspace{1cm} (B-11)
10.4 Axial Induced Velocity

The figures B-23 and B-24 in this section show the axial induction factor in a plane at 0.05 times the rotor diameter behind the rotor disc. The induction factor $a$ is the ratio of the local induced flow velocity to the free-stream flow velocity, as given in equation B-11.

Figure B-23: Axial induction factor at 0.05D behind the rotor plane for axial inflow

Figure B-24: Axial induction factor at 0.05D behind the rotor plane for yawed inflow (15°)
10. Simulation results

10.5 Wake Visualization

In order to investigate the flow behavior in the near wake, the wake shape is plotted in figures B-25 and B-26. All plots are of the $15^\circ$ yawed inflow case. In these plots, all dimensions are expressed relative to the rotor radius.

(a) Wake Length: $\frac{1}{2}$ rotation

(b) Wake Length: 1 rotation

Figure B-25: Visualization of the vortex sheet in yawed inflow conditions

(a) Sideview

(b) Topview

Figure B-26: Visualization of the vortex sheet in yawed inflow conditions
PART B. RIGID GEOMETRY SIMULATION

10.6 Pressure Distribution

The 2D pressure distribution over the blade at 75% of the radius is plotted in figure 28(a). As a reference, the pressure distribution obtained with XFOIL is shown in figure 28(b). The scaling on the y-axis of both plots is left intentionally blank, as the quantitative values are very hard to compare. The reason for this is found in the definition of the pressure coefficient, which can be found in equation B-12. The $C_p$'s obtained from different sources use different values for the reference velocity $V_\infty$. The most important conclusion to be drawn from the general shape of the pressure distributions is the inability of the panel code to handle flow separation, while the results from XFOIL clearly show the occurrence of stalled flow over more than half of the airfoil section, even with a transition point near the leading edge of the airfoil.

\[
C_p = 1 - \frac{V^2}{V_\infty^2} \tag{B-12}
\]

Figure B-27: Pressure coefficient of the blade section at 75% radius
10. Simulation results

(a) Panel code (Axial inflow)  
(b) XFOIL (2D) ($\alpha = 18^\circ$)

Figure B-28: Pressure coefficient of the blade section at 75% radius
Part C

Flaps Affecting Blade Aerodynamics
11 Flap Geometry

Although the flaps designed for the practical wind tunnel model consist of two separate flap structures per blade, the flapping motion in the simulation is implemented using one single flap per blade. The dimensions of the single simulated flap are the same as the summation of the two separate flaps used in real life. Both flap configurations are shown in the blade planforms in figure C-1. Both real flaps operate synchronous, having the same deflection angle at all times. The main difference between the real and the simulated case is therefore the distance of the flap with respect to the rotational axis of the rotor. As a result, the exact values of the loads at the blade root might slightly differ. For the purpose of investigating the general effects of adding control surfaces to the rotor blades, a single flap is assumed to be appropriate. In the future, it might be a possibility to change the way the flap geometry is defined in order to include two separate flaps.

![Real blade planform](image1.png)  ![Simulated blade planform](image2.png)

Figure C-1: Blade layout including flap positions (shaded)

12 Integration in the Simulation

The original simulation structure did not directly include changes in the geometry during the simulation. It merely created an infinitely stiff structure at the start of the simulation, which was rotated the appropriate azimuth step every time-step of the simulation. The new program however had to incorporate an update of the geometry within every time-step. It should renew the position and orientation of the panels which are part of the control surface. As a result, several time-consuming calculations had to be performed every step instead of only once at the beginning of the simulation. It was also not possible anymore to recalculate the blade geometry from the initial data. Therefore, the geometry data had to be stored every time-step in separate variables, in order to be able to reconstruct the simulation in postprocessing. A simplified version of the flow diagram of the simulation procedure (figure A-3 on page 15) is shown in figure C-2. There are three dark-blue boxes within
PART C. FLAPS AFFECTING BLADE AERODYNAMICS

this flow diagram, which emphasize the flap deflection procedure. First, during the initialization of the simulation, the indices of those points within VertPts which are part of the flaps are marked. This way, during the simulation, only those points can be rotated around the local hinge-line. After that, the whole blade is rotated to its required azimuthal position.

Figure C-2: Flow diagram of the flap deflection procedure in the simulation

13 Controlled Flap Deflection

For this first phase, the flap deflection is prescribed before starting the simulation. Of course, the later purpose suggests that the flap deflection angles should be determined in realtime. In order to do that, it is very useful to know in advance the effects of the flaps. There are two separate reasons for that.

The first reason is that it might be very useful to evaluate the influence of the flap deflections on different variables. These variables can for instance be the power output or the root bending moments. This way it becomes possible to determine which variable is most suited to be optimized using the flap deflections.

The second reason is knowing what the relation is between a given flap deflection and the changes of the variable of interest. The relation between flap deflection and variable change has a dual cause–and–effect behavior. Not only has the amplitude of the flap deflection angle an effect on the amplitude of the output, also a possible phase difference can be present between the cause and the effect.

To investigate the effects, a simple prescribed flap deflection mode is chosen: a pure sinusoidal cycle. This deflection cycle is characterized by the amplitude and the period of the cycle. As an initial value, an amplitude of 3° is assigned, and for convenience, the period of the flap deflection is chosen to be equal to 1P. I.e. the time of one flap cycle is exactly the same as the time needed for one rotor rotation.

Of course, it is beneficial to study the effects of the flaps as an isolated phenomenon. Thus the flow case which is started from, is the pure axial inflow case. This way, all 1P effects can be assumed to originate from the 1P flap deflection cycle.

The flap deflection angle is plotted with respect to blade azimuthal position in figure
C-3. The deflection cycle plotted in figure C-3 is the same for both rotor-blades.

![Flap deflection cycle](image)

Figure C-3: Flap deflection cycle (purely sinusoidal with 1P period and 3\(^\circ\) amplitude)

In order to get a feeling of the influence of the amplitude, similar analysis is done with a flap amplitude of 1 and 6 degrees.
14 Effects of Flap Control

The different variables of interest can be plotted in the same way as they were plotted in part B. Since the polar plots of the distributed loads along blade span only provide a general overview of the load behavior, they are not given here for comparison of the flap deflections. The most clear effects of the flap deflection modes are visible in the total loads at the blade roots. Therefore only these plots are given in this section.

- **Flapping Moment Coefficient**

  ![Flapping Moment Coefficients](image)

  (a) No flap deflection  
  (b) Flap deflection amplitude 3°

  Figure C-4: Root flap-wise bending moment (non-dimensional) vs. azimuth

- **Pitching Moment Coefficient**

  ![Pitching Moment Coefficients](image)

  (a) No flap deflection  
  (b) Flap deflection amplitude 3°

  Figure C-5: Root pitching moment (non-dimensional) vs. azimuth
14. Effects of Flap Control

- Thrust Coefficient

![Figure C-6](image)

(a) No flap deflection

(b) Flap deflection amplitude $3^\circ$

Figure C-6: Rotor Thrust vs. azimuth (contribution of one single blade)

![Figure C-7](image)

(a) No flap deflection

(b) Flap deflection amplitude $3^\circ$

Figure C-7: Rotor Thrust vs. azimuth (contribution of the total rotor)

In figures C-4, C-5 and C-6, the effect on the root flapping moment, the root pitching moment and the thrust contribution of one single blade is clearly visible. The effects are produced with a flap deflection according to the profile in figure C-3 with a $3^\circ$ amplitude. It appears that there is no phase shift in the flapping moment and the thrust, but for the pitching moment, a small phase shift can be observed of about $-10^\circ$. The reduced frequency might help to discover the origin of this small phase shift. The reduced frequency of the flapping motion is 0.066.
PART C. Flaps Affecting Blade Aerodynamics

In the plots of the rotor torque and power (figures C-8, C-9 and C-10 respectively) the effect of the same flap deflections is hardly present, and almost unrecognizable as a sinusoidal behavior. Knowing this, it can be concluded that it would be unwise to try and control some unwanted behavior in the power output by applying flaps. The beneficial effect on the power would be overshadowed by an immense unwanted effect on the blade moments and the rotor thrust.

- **Torque Coefficient**

(a) No flap deflection

(b) Flap deflection amplitude 3°

Figure C-8: Rotor torque vs. azimuth (contribution of one single blade)

- **Power Coefficient**

(a) No flap deflection

(b) Flap deflection amplitude 3°

Figure C-9: Rotor Power vs. azimuth (contribution of one single blade)
15 Effects of Flap Amplitude

The effect on the power and thrust of the different flap amplitudes is shown in figures C-11, C-12 and C-13. It should come as no surprise that the shapes shown in figures C-11 and C-12 are more or less the same for the different flap amplitudes. Also, the amplitudes of the response change according to the applied flap angle. To show the relation between the flap amplitude and the response amplitude, the standard deviations from the mean are plotted in figure C-13. The standard deviations of both the power and the thrust have a clear linear behavior with respect to the applied flap amplitude.

Figure C-11: Rotor Power vs. azimuth (contribution of one single blade)
PART C. FLAPS AFFECTING BLADE AERODYNAMICS

(a) Flap deflection amplitude 1° (b) Flap deflection amplitude 3° (c) Flap deflection amplitude 6°

Figure C-12: Rotor Thrust vs. azimuth (contribution of one single blade)

(a) Power (b) Thrust

Figure C-13: Standard deviation for different flap amplitudes
The induced velocity due to the rotor is directly related to the trust force via Newton’s action-reaction principle. The rotor thrust is one of the rotor characteristics that are strongly influenced by the flap deflections. Therefore it is useful to also look at the induced velocity when investigating the effect of the flaps. In figure C-14, the induction factor is plotted in a fixed point closely in front of the rotor disc at azimuth of 90°. The induction factor is calculated from the induced velocity according to equation B-11 on page 37. The factors plotted in figure C-14 are the contributions of the wake only. As reference, the total induction factor (including the rotor-blade) is plotted as the red dotted line in order to have a clear indication of the moment at which the blade passes the measuring point. The since the effect of the wake is isolated, there is hardly any change in the induction factor for the first quarter of the rotation. The wake-panels present behind the rotor are the remainders of the previous rotations without the flap deflections. When the first blade passes at 90°, the effect of the flap movement becomes visible in the representation of the induction factors. When the second blade passes the measuring point (270°), the flap effect is increased due to the presence of the ‘flapped’ wake of the first blade passage.

Figure C-14: Induction factor at a fixed point vs azimuthal position of the rotor. Note that positive induction is slowing down the incoming wind.
16 Applicability of Smart Actuators During Yaw Misalignment

One of the reasons on why a turbine should be equipped with flaps is the goal to reduce the cyclic loadings in order to increase the fatigue lifetime of the turbine blades. Another possibility is to level out unwanted behavior in the power output. In part B, it was concluded that the effect of yaw misalignment was mainly sensed in the torque and the power, which showed a strong cyclic behavior. On the other hand, in section 14 it is shown that the flap deflection shows very little influence on the torque and the power output, while the rotor thrust and flapping moment are strongly subjected to periodic effects.

These trends give strong indications that the application of flaps might not be suited to handle the periodic effects caused by yaw misalignments. Using the simple velocity and force vector system in figure C-15, a clarification is given to what the causes are for the aforementioned phenomena.

As shown in figure C-15, the lift and drag of an airfoil section are always expressed perpendicular and parallel to the local flow velocity. The addition of both force vectors is the resultant force vector plotted in blue. If the Force resultant is tilted forward (only very slightly in figure C-15), this airfoil section contributes to the rotor torque and thus also the power.
16. Applicability of Smart Actuators During Yaw Misalignment

- Lets first consider a two-bladed rotor experiencing a case of Yaw misalignment. The blade that moves into the wind direction will experience a slight increase in local flow velocity and at the same time a reduction of the inflow angle. The blade moving with the wind direction is subjected to a slight decrease of the local flow velocity and an increase of the inflow angle. Looking only at the blade which moves with the wind, it should be noted that with the increased inflow angle, the whole system of local velocity, lift, drag and resultant forces rotates around the aerodynamic center against the clock. As a result, the force resultant will be directed more forward, causing a strong change in the rotor torque and the power even when there is only a slight change of the inflow angle.

The second effect of the increased inflow angle is an increase of the local angle of attack. This results in a quantitative increase of both the lift and the drag force. However, this increase of the total resulting force is canceled by the simultaneous decrease of the local inflow velocity. For the other blade, moving into the wind direction, the same reasoning will of course yield the opposite result.

Thus, the effect of the yaw-misalignment is clearly visible in the torque and the power, while the effect on the thrust and flap-wise bending moment is only marginal.

- When investigating the effect of a Flap deflection, it should be noted that none of the vectors in figure C-15 changes its orientation, which is the effect which provides the characteristics of the yaw case. Instead, a downward deflection of the flap can be seen as a increase of the camber of the airfoil and simultaneously a clockwise rotation of the whole camberline, increasing the local angle of attack of the airfoil section. Both aforementioned effects result in an increase of both the lift and the drag. As both increase, the orientation of the resultant force may stay more or less the same, and only its quantitative value increases. Similarly, an upward deflection of the flap will result in a decrease of the thrust and flapping moments.

This automatically implies that the thrust and the flapping moment will experience an reasonable influence of the flap movements, while the torque and power remain more or less unaffected.
17 General Applicability of Smart Actuators

Although the application of flaps with smart control seems rather disappointing according to the above reasoning, there is hope for the application to succeed in situations with unwanted disturbances other than yaw misalignment. When the results given by Nengsheng Bao et al. in [1] are considered, the effect of the flap deflection on the torque contribution is expected to be reasonable. For non-stalled conditions, this is the case because the change of the lift coefficient is a lot stronger than the change of the drag coefficient. With relatively small angles of attack, there is a side-effect. In this case, the axial force (thrust) mainly consists of the lift vector, and is therefore even more influenced by the flap deflection than the torque. This confirms the presumption that the it might not be wise to use flaps to influence power fluctuations, since it would result in large unwanted variations of the rotor thrust and the flapping moment.

When operating under large angles of attack, the lift vector is oriented more forward, and the influence of the changing lift force has more influence on the torque. But, as can also be concluded from the results in [1], the change of the drag as a result of the flap change also becomes very strong, reducing the effect on the torque.
Part D

Influence of Wake Cut-off
18 Limiting Simulation Time

In almost every form of simulation the same trend can be observed: The more advanced the simulation which tries to approximate the physical reality, the more time it takes to finish the simulation. Less assumptions means more variables, and thus more required computational power and time. A more refined spacial discretization means even more required computational power and time.

In order to reduce the time it takes to finish one simulation, it is useful to know to what extent certain simulation variables effect the obtained simulation results, and to what extend they influence the simulation time.

19 Practical Application

Within the used free wake panel code simulation, the calculation of the induced velocity is a time-consuming part. The cause for this high time-consumption is the fact that in order to update the free wake, the local flow velocity for every point in the wake has to be calculated. This means that for every wake-point the induced velocity contributions of all blade and wake-panels have to be calculated. The number of panels in the blade stays the same throughout the simulation, but the number of wake-panels to include increases with every time-step. If the number of included wake panels can be limited, the needed simulation-time per time-step will hardly increase once this limiting number of time-steps is reached, where it would otherwise keep increasing every time-step. This way, the total calculation time can be dramatically reduced.

But in order to do so, the effect of the reduced number of included panels on the obtained induced velocity should be known. With this in mind, the induced velocity is calculated at the rotor plane for different wake cut-off lengths. The induced velocity components provided by the blade panels are not taken into account for the plots in this part. The contribution of the blades is of course much stronger than the contribution of the wake panels, and would therefore strongly reduce the visibility of the wake influence.
PART D. INFLUENCE OF WAKE CUT-OFF

20 Results

The effect of different wake cut-off lengths on the induced velocity is shown figure D-2. As is the case for figure C-14 on page 53, the induced velocity is calculated in a fixed point at an azimuth of 90° just ahead of the rotor-plane. Also in this case, the total induced velocity including the blades is shown as a red dotted line to indicate the passage of the blades. This total induced velocity is obtained using 7 rotations of wake-steps, so it corresponds to the black line indicating the induced velocity without the contributions of the blades.

As can be seen, the induced velocity calculations using only a small part of the wake (90°, 180° and 1 revolution) give severe underestimates of the real value. The minimal difference between the calculation including seven rotations and the one using three and a half rotation suggests that the seven rotations case will closely reflect the exact solution with a virtually infinite wake.

To answer the main question in this wake cut-off analysis, a simple plot is made of the mean values of the induced velocity for different wake cut-off lengths. It can be seen that when three or less rotations are used to calculate the induced velocities during the simulation, no sufficiently accurate result will be produced. However, using four or more rotations will result in a fairly precise result, providing a good representation of the case with a fully developed wake.

As can be seen, the induced velocity calculations using only a small part of the wake (90°, 180° and 1 revolution) give severe underestimates of the real value. The minimal difference between the calculation including seven rotations and the one using three and a half rotation suggests that the seven rotations case will closely reflect the exact solution with a virtually infinite wake.

To answer the main question in this wake cut-off analysis, a simple plot is made of the mean values of the induced velocity for different wake cut-off lengths. It can be seen that when three or less rotations are used to calculate the induced velocities during the simulation, no sufficiently accurate result will be produced. However, using four or more rotations will result in a fairly precise result, providing a good representation of the case with a fully developed wake.

Figure D-1: Mean value of the induction factor for different wake cut-off lengths. Note that the blue circles are the values plotted in figure D-2.
Figure D-2: Induction factor at a fixed point vs azimuthal position of the rotor. Note that positive induction is slowing down the incoming wind.
Part E

Conclusions
Simulation

- **Flow Angles**
  The combination of tip speed ratio (4) and a geometry with relatively small twist angles and no pitch results in very high angles of attack over the entire blade span. These angles of attack increase from 14° at the tip to 37° at the root sections.

- **Stall**
  From the analysis of a 2D airfoil section using XFOIL, it appears that even the blade section at the tip would experience severe stall conditions during normal wind velocities. The 2D flow separation at the tip section is predicted to occur around mid-chord.

- **Panel Method Suitability**
  Although the panel method is judged as a very suitable means to perform the simulation, its validity for this particular rotor configuration should be questioned. The reason is the method’s inability to handle the strong flow separations occurring. The tests of the rotor-configuration in the Open Jet Wind Tunnel Facility of the Delft University of Technology can serve as a reference. There it can be investigated to what extend the stall behavior is correctly predicted, and a comparison can be made with the results from the panel code.

- **Negative Loading Near Blade Root**
  As is explained in section 10.2, the span-wise distributions of force and moment coefficients obtained from the simulation unexpectedly show a common trend near the root section of the blades. The loading dramatically reduces to even negative values near the blade root.
  This phenomenon can be explained as a combination of several 3D effects: the presence of a large local chord length and a high twist angle at sections very close to the root produce the unexpected behavior. The obtained values are not wrong, but appear to be because of the way the panel contributions of the forces are added up. Panels are added up according to their span-wise position as they form the airfoil cross-sections. The panels of one selected airfoil section near the root can have the same span-wise position, but certainly do not experience the same radial distance from the rotor axis. The combination of a large chord and a small span-wise position enhance this phenomenon. It should be possible to add up the panels according to concentric circles, combining panels which all have the same radial distance. However, for this change to work, it would take a complex redistribution of the panels, severely complicating the blade construction at the start of a simulation.
Influence of Yaw Misalignment

When a wind turbine rotor is operating in a condition of yaw misalignment, the blades experience a periodic variation of the inflow angle and thus angle of attack and the local inflow velocity. Lift and drag are expressed as force components perpendicular and parallel to the local inflow respectively. This means that with the periodically changing inflow angle, the orientation of the resultant vector of both the lift and the drag changes back and forth. This pending motion of the force resultant has a strong influence on the torque and thus the power contribution, but leaves the thrust force on the rotor more or less unaffected. The second effect of the change in inflow angle is the equal variation of the angle of attack. This has a periodic effect on the loads. But as initially stated, also the local inflow velocity fluctuates. The increase of angle of attack is always combined with a decrease of the velocity, and vice versa. These effect more or less cancel out, and the final variation of the thrust and flapping moments are very small.

Influence of Flap Deflections

In a similar fashion to the previous conclusion, it is possible to analyze the effect of a cyclic flap deflection while the rotor is operating in pure axial inflow conditions. In this case, the local inflow angle and the orientation of the lift and drag vectors do not change. The deflection of the flap can be seen as a combination of a change of the airfoil camber and a rotation of the airfoils chord line. Each contribution leads to an increase of both the lift and the drag. As the magnitude of the resultant force vector changes, the thrust and flapping moments are directly influenced. At the same time, the orientation of the resultant force remains more or less the same which means the torque and power are hardly affected by the flap deflections.

Applicability of Flaps on Wind Turbine Blades

As a result of the difference between the effects of yaw misalignment and the effects of flap deflections, it seems in vain to add flaps to a wind turbine blade. When an even amount of blades is used, the fluctuations in the power is canceled. But in the case of an odd number of blades, the power output will still show a cyclic behavior. It would indeed be unpractical to use flaps in order to level this power output.

On the other hand, if it is the objective to increase the fatigue life or the weight of the blades, it would be useful to level the root flapping moment instead of the power. In the case of yaw misalignment, there is hardly any cyclic behavior of the flapping moment, and thus very small flap deflections would be required.

Of course there are also other phenomena which can result in unwanted blade loading, like windshear, turbulence and gust. Especially in the case of a gust it can be expected that the all loads are suddenly increased and decreased. For this case,
it would clearly be beneficial to use flap deflections to keep the flapping moments under control. The sudden increase in power which also accompanies the wind gust, however, will hardly be affected by the control operations.

The blades flap-wise bending moment can be controlled with the application of flaps. Power should not be tried to control, as this would resulting in extreme flap-wise bending moment and thrust force variations.
Part F

What’s Next?
Aero-Elastic Analysis
21 Basics of Aero-Elasticity

As the term suggests, an aero-elastic analysis is a combination of an aerodynamic part and an elastic part. An aerodynamic model has the possibility to calculate resulting forces as an effect of airflow around a body. An elastic model is able to calculate the structural deformations of an object under a given loading. With this in mind, it is easy to come up with the idea of coupling both models. The aerodynamic model calculates the forces as a result of the airflow around the body and the elastic model calculates the changes in geometry of the body subjected to those forces. The changed geometry shape will change the airflow and the aerodynamic forces need to be recalculated.

As wind turbines become larger, blades become more slender, and tend to be more and more subjected to structural deformations under aerodynamic loading.

Some of the effects have to be considered during the design, like the flap-wise bending of the blades which might result in collision of the rotor blades with the tower structure.

Other effects can be taken into account, but will occur nonetheless. This can e.g. be the loss in power due to a reduced angle of attack as a result of blade torsional deformation.

But the most important aero-elastic phenomenon is flutter, which is generally always destructive. Flutter is a well known problem for aircraft with slender wings and is an extreme case of structural resonance under cyclic aerodynamic loading. It literally wouldn’t be an aero-elastic effect if this cyclic aerodynamic loading was not the result of the structural resonance. This phenomenon is most occurring as a cycle of angle of attack, lift and bending moment, as given in figure F-1. As mentioned in the figure, flutter is often initiated by a change in pitching moment with a resulting

---

Figure F-1: Flutter cycle
PART F. THE NEXT STEP: AERO-ELASTIC ANALYSIS

torsional deflection increasing or decreasing the angle of attack. This triggering change of the pitching moment can easily be an accidental side-effect caused by the deflection of a control surface.

Wind turbine blades can be seen as fairly slender wings, and thus suffer the same risk of flutter occurrence. When flaps are added to these blades, the risk greatly increases. Especially when automated flap controllers can unintentionally cause or even invigorate the flutter.

22 Aero-Elastic Simulation

In order to avoid the above described aero-elastic effects, it is very useful to have a model which incorporates not only aerodynamic loading on the turbine blades, but also includes the effects of the structural deformations resulting from these loads. For this purpose, the aerodynamic simulation code used in the previous parts of this thesis is to be combined with a structural model. The structural model selected for this job is an existing model, produced by Thanasis Barlas at the Delft University of Technology. It is a super-element model built using the SimMecanics solver in Simulink. Since Simulink is related to MATLAB, the interaction between both models was assumed to be straightforward. The actual connection, however, proved to be a lot more difficult, and although the main setup of the total program is realized, the actual execution requires further troubleshooting. Finding and solving bugs in the simulation is a time-consuming activity, and a clear time-estimation for the finishing of this code is very difficult, as it depends on the unforeseeable amount of bugs left in the simulation program. The flow diagram of the coupled simulation as it is set-up currently is shown in figure F-2.

![Flow Diagram of the Aero-elastic model](image)

Figure F-2: Flow Diagram of the Aero-elastic model
Bibliography


